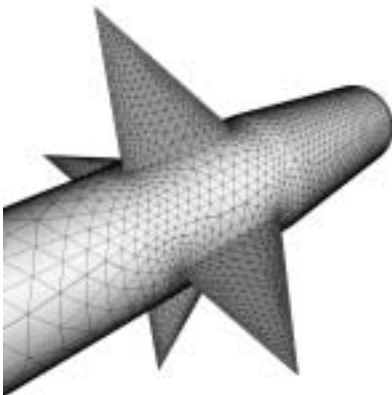


## Computational Fluid Dynamics

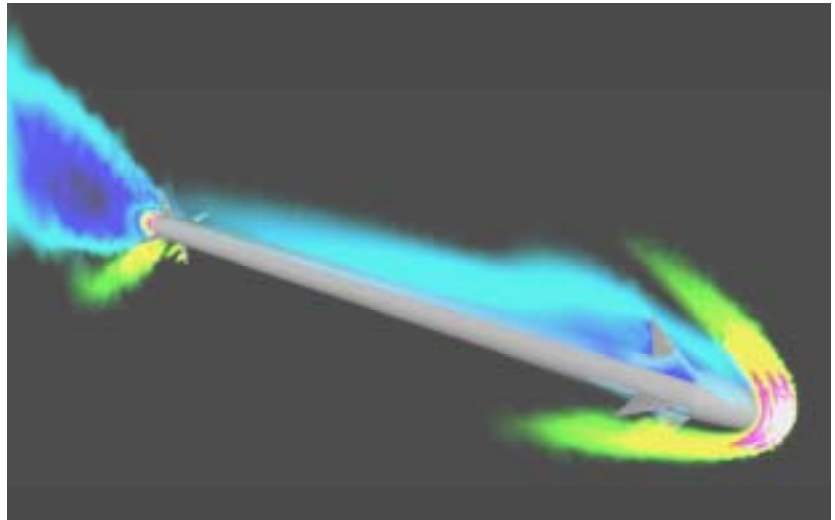
***Computational Fluid Dynamics or CFD*** is a powerful technique to investigate fluid flow, heat transfer, or other physics – such as chemical reactions – using computer-based simulation. Some have even suggested calling CFD techniques a ‘digital’ wind tunnel to highlight the computer’s ability to model wind tunnel, or even flight test, type experiments. A CFD simulation involves

- surface and volume mesh generation
- numerical solution over domain of interest
- numerical solution visualization
- analysis of results/comparison with experimental data

Commercial software packages as well as government, faculty and student developed codes are being used in the department to numerically investigate aerodynamic flow fields.



Unstructured Mesh about  
Air-to-Air Missile Nose



Air-to-Air Missile with Solid Rocket  
Motor Firing CFD Solution

***For further information or to suggest a related thesis topic, please contact:***

Lieutenant Colonel Montgomery C. Hughson  
Assistant Professor and Deputy Department Head  
(937) 255-3636 x4597 (DSN 785)  
[Montgomery.Hughson@afit.edu](mailto:Montgomery.Hughson@afit.edu)